Imperial College London

A meshing tutorial

Fluidity training event

Applied Modelling and Computation Group

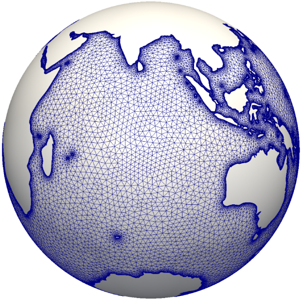
Department of Earth Science and Engineering, Imperial College London

5-9 November 2012

AMCG

Tutorial overview

- What is a mesh.
- What is Gmsh.
- Viewing and meshing a 3–D geometry.
- Generating and meshing a 2–D geometry.
- Meshing realistic domains.



What is a mesh?

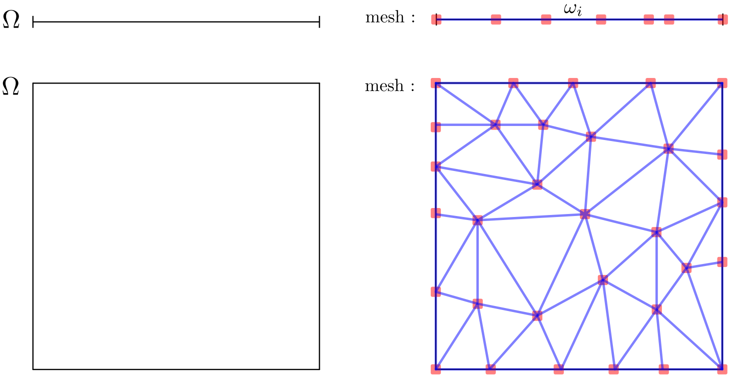
A mesh can be qualitatively thought of as the tessellation of a domain Ω into a set of non-overlapping sub-domains ω_i:

$$\begin{aligned} \Omega &= \cup \{\omega_i \,|\, i = 1, 2, \ldots ele\} \\ 0 &= \cap \{\omega_i \,|\, i = 1, 2, \ldots ele\} \end{aligned} \tag{1}$$

where *ele* is the number of elements in the tessellation.

What is a mesh?

A mesh can be qualitatively thought of as the tessellation of a domain Ω into a set of non-overlapping sub-domains ω_i:



What is Gmsh?

- It is the role of the mesh (or grid) generator to scatter the points and generate the mesh, whilst ensuring high quality elements.
- Gmsh is a "3D finite element grid generator with a build-in CAD engine and post-processor. Its design goal is to provide a fast, light and user-friendly meshing tool with parametric input and advanced visualization capabilities."[1]. Furthermore, Gmsh can be used as a 1–, 2– and 3– dimensional mesh generator for use with the Fluidity CFD code.
- Not the only mesh generator that can be used with Fluidity.
- Distributed under the GNU General Public License, available for Linux, Windows and Mac OS.

[1] from `http://www.geuz.org/gmsh/`

Starting Gmsh

Open up a terminal and issue the following commands:
wget amcg.ese.ic.ac.uk/files/annulus.geo (to fetch an example-file from our servers)
gmsh annulus.geo (To open the example-file with Gmsh)

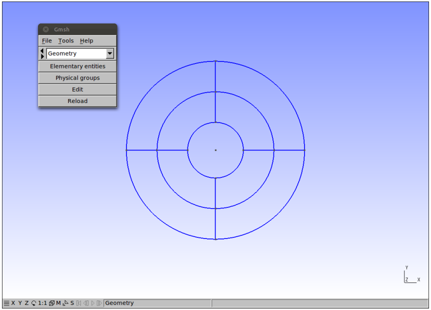


Figure: Gmsh startup with default view on a 3–D annulus

Navigating menus, Manipulating view

- Navigating menus
  - Gmsh's architecture is centred around four modules.
  - Geometry (G), Mesh (M), Solver (S), Post–Processing (P).
- Manipulating views (mouse)
  - Panning : Hold right button down and move cursor.
  - Zooming : Scroll or hold middle button down and move cursor.
  - Rotating: Hold left button down and move cursor.
- Meshing the annulus
  - Mesh (M) $>$ 3D

A two dimensional example

- To generate a 2–D, structured mesh on a $1,000$ km $\times 1,000$km square:

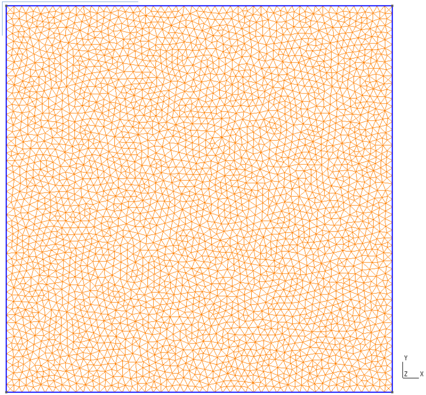


Figure: Meshed surface of 2–d example.

Getting started

From a terminal, start Gmsh using:

gmsh 2d−example.geo

We proceed by defining:

1. Points
2. Lines
3. Surfaces

Setting up the geometry: Points

- Geometry (G) > Elementary Entities > Add > New > Point
- The Contextual Geometry Definitions window will appear.
- Enter the point coordinates and click "Apply".
- Do not move cursor outside the Contextual Geometry Definitions window while entering the coordinates!
    - Point 1: [0., 0., 0.] with characteristic length of 2e4
    - Point 2: [0., 1.e6, 0.] with characteristic length of 2e4
    - Point 3: [1.e6, 1.e6, 0.] with characteristic length of 2e4
    - Point 4: [1.e6, 0., 0.] with characteristic length of 2e4
- Press 'q' and close Contextual Geometry Definitions window.

Setting up the geometry: Lines

- Geometry > Elementary Entities > Add > New > Straight Line
  - Join two points by selecting them.
  - Join points (1,2), (2,3), (3,4), (4,1), in that order please!
  - Press ‘q’

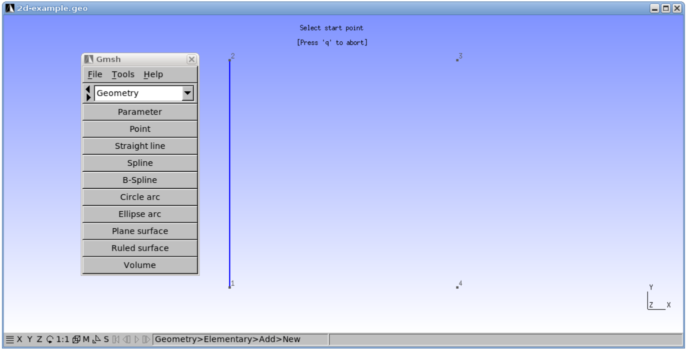


Figure: The Gmsh windows while drawing lines & showing the point ID’s.

Setting up the geometry: Declaring a plane surface.

- Geometry(G)>Elementary Entities >Add >New >Plane Surface
- Click on any of the sides.
- Press 'e' then 'q'.

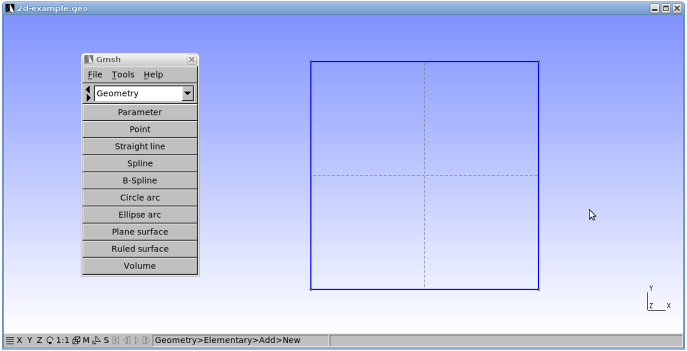


Figure: The Gmsh windows showing the Plane Surface highlighted by grey, dash-lines.

Physical groups: Lines and Surfaces

In order to specify regions and boundaries in Fluidity, they must first be defined as “Physical Groups” in Gmsh:

- Assign “Physical Line” ID’s to the domain boundaries
  - Geometry (G) > Physical Groups > Add > Line
  - Select bottom side and press ‘e’.
  - Repeat for right, top and left sides, in that order!
- Assign “Physical Surface” ID to the plane surface
  - Geometry (G) > Physical Groups > Add > Surface
  - Select the highlighted surface and press ’e’ on your keyboard.

Producing a mesh

Finally, to produce a 2–D mesh: Mesh (M) > 2D

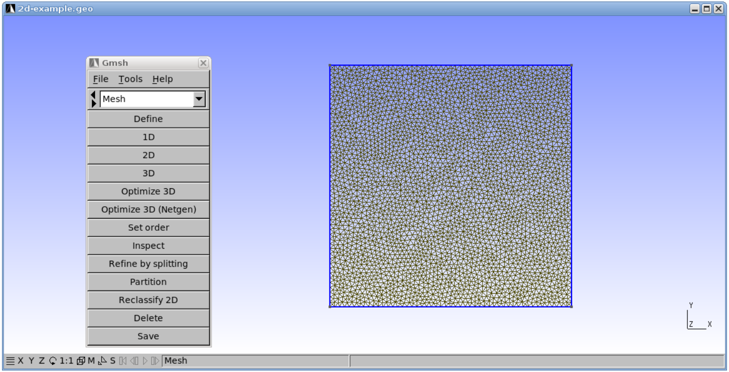


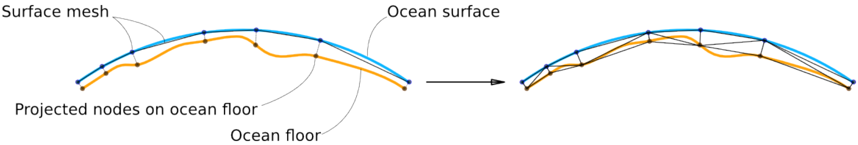
Figure: The Gmsh windows showing the mesh for this example.

Meshing realistic domains – Our approach so far

Domain geometry & Mesh are constructed on a spherical shell, representative of the Earth's surface geoid.

Tutorial `http://amcg.ese.ic.ac.uk/files/gmsh_tutorial.pdf` or `http://perso.uclouvain.be/jonathan.lambrechts/gmsh_ocean/`

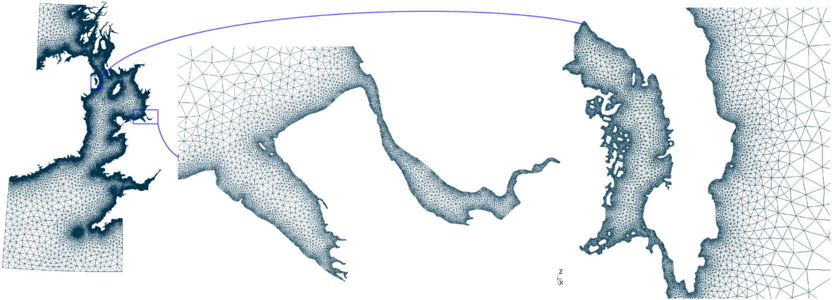
- Coastlines extracted from GSHHS dataset, via Gmsh plugin.
- Open boundaries are drawn, usually constant longitude & constant latitude lines.
- Element size is defined using "attractor" and "threshold" fields available in Gmsh.
- 2–D surface mesh is extruded along the radial direction within Fluidity.



Shortcomings – Domain boundaries.

A simple CAD engine is insufficient:

- Shorelines are geometrically very complex.
- The Gmsh GSHHS plugin fits a spline through the GSHHS points, leading to intersecting shorelines.
- Drawing arbitrary lines as open boundaries –e.g. contour at a given depth– not easily done.



Shortcomings – Mesh size

The spatial variation of the mesh element size could be complex:

- Mesh element size must usually be fine near the coastlines to capture their structure.
- Mesh element size must usually be fine in areas of steep ocean floor topography and in shallow areas: Not easily done with current approach.

Meshing realistic domains – Our proposed approach

Use Geographical Information Systems to extract domain boundaries and prescribe mesh metric size.

- Existing GIS software capable of reading databases in popular formats.
- Capability of extracting contours from field-type databases is usually available. ($\rightarrow$domain boundaries)
- Capability of generating field-type databases is usually available. ($\rightarrow$mesh size metric)
- A project bringing together GIS software with mesh generation software has been registered by AMCG:
  - `https://launchpad.net/meshing`
  - Publication coming soon.
  - Watch this space!

Further reading

- A Gmsh tutorial AMCG
- C. Geuzaine and J.-F. Remacle, Gmsh: a three-dimensional finite element mesh generator with built-in pre- and post-processing facilities.. International Journal for Numerical Methods in Engineering, Volume 79, Issue 11, pages 1309-1331, 2009.
- C. Geuzaine and J.-F. Remacle, Gmsh Reference Manual.. Available at, `http://geuz.org/gmsh/#Documentation`.
- A. Avdis and S.L.Mouradian, A Gmsh tutorial, available at, `http://amcg.ese.ic.ac.uk/files/amcg-gmsh-tutorial.pdf` .